

CNC EXPERT STANDARDS

PROJECT: TITAN-1M
CERTIFICATION: CAD

OVERVIEW:

Successfully demonstrated comprehension of the TITAN-1M CAD performance standard. Create an accurate and fully defined TITAN-1M solid model using CAD software and part documentation.

MATERIALS:

- TITAN-1M part print
- Computer with CAD software

SKILLS DEMONSTRATED:

SKETCHING

- **Create Sketch:** Enters Sketch mode and creates a new sketch on a selected plane or face.
- **Select Construction Plane:** Selects plane on which to sketch part dimensions.
- **Two Point Rectangle:** Creates a rectangle using two points for the diagonal corners.
- **Center Diameter Circle:** Creates a circle using a center point and diameter.
- **Line:** Creates lines and arcs.
- **Point:** Creates a sketch point.
- **2D Fillet:** Places an arc of a specified radius at the intersection of two lines or arcs.
- **Trim:** Trims a sketch curve to the nearest intersecting curve or boundary geometry.
- **Offset:** Copies the selected sketch geometries a specified distance from the original geometries.
- **Dimensioning:** Assigns dimension values to sketch geometries and/or modifies existing dimensions.

CREATING

- **Hole:** Creates a hole based on user-specified values and selections.
- **Thread:** Add internal or external threads to cylindrical geometry.

MODIFYING

- **Press Pull:** Modifies the selected geometry using offset, extrude, or fillet commands.
- **3D Fillet:** Places an arc of a specified radius at the intersection of two lines or arcs.